

# 2D-Image to 3D-STL

<b>Project</b>	2dimage-to-3dstl
<b>Description</b>	Convert a 2D-Image to an 3D-STL for your 3D-Printer
<b>Status</b>	completed
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<b>Participants:</b>	/

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## What do you need

- An image, preferably SVG or at least a high-res version of it.
- A PC, preferably with Linux installed. ;)
- An idea of what you want to do with your STL later on
- Internet

## Some basic Info

You will learn, how to turn your 2D Logo/Image into a 3D STL, which you can use in most other 3D-CAD programs. Yes I wrote „most other“ and not „all“ 3D CAD program. You might ask yourself now, why I did that but there is a very simple reason. There are 2 STL versions, one is build on a binary core, the other one on an ASCII core. But fear not, you will learn in this guide, a way where it doesn't matter witch STL-version there is and you will see it work on an example with lots of pictures.

The way your going to learn is tested and verified on an Ubuntu 15.10 OS, I will announce, but not explain, a few other methods which would work but didn't work for me because of multiple reasons. Thanks to our member [orimpe](#) and some guidance from the C3L it was possible to get to the mark, so a small donation, as thanks, would be nice and very much appreciated. To make it more attractive, take a look at our donations gifts.

## Prepare you PC

To convert an 2D logo into an 3D STL, your going to need to install the program Inkscape, OpenSCAD, an CAD like freecad, an account for tinkercad and an extension for Inkscape. All those things don't cost a thing.

Inkscape

**Inkscape – Vektorgrafikbearbeitung**  
Skalierbare Vektorgrafiken (SVG) erzeugen und bearbeiten  
★★★★★ (403 Bewertungen)

**Installiert** Entfernen

Inkscape is an illustration editor which has everything needed to create professional-quality computer art. You can use it to make diagrams and illustrations, technical drawings, web graphics, clip art, icons and logos. A collection of hands-on tutorials show you how to combine lines, shapes and text of different types and styles to build up a picture.

A selection of powerful vector graphics editing tools comes as standard. There is excellent support for paths, gradients, layers, alpha transparency and text flow control. An extensive library of filters allow you to apply realistic effects and extensions allow you to work with bitmaps, barcodes and printing marks, amongst other things.

Most of the common vector formats are supported, including PDF, Adobe Illustrator and AutoCAD files, and it has unrivalled support for the SVG web graphics standard.

[Entwicklerwebseite](#)

**Zusätzliche Erweiterungen**

- GNOME »Virtual File System« (Zusatzmodule) (libgnomevfs2-extra)
- Konvertierungsprogramme für das Windows-Metafile-Format (libwmf-bin)
- Perl interface to ImageMagick – transition package (perlmagick)
- Hilfsprogramme zum Konvertieren von Dateien mit XFig-Abbildungen (transfig)
- Diagrammbearbeitung (für GNOME) (dia-gnome) ★★★★★
- Perl module to generate SVG images (libsvg-perl)
- Perl-Modul für XML-Baumstruktur-Abfragen mittels XQL (libxml-xql-perl)
- Konverter von PostScript und PDF in editierbare Vektorgrafiken (pstoedit)
- Universal vector graphics translator (python-uniconvertor) ★★★★★
- Interpreter für die objektorientierte Skriptsprache Ruby (Standardversion) (ruby) ★★★★★

## OpenSCAD

**OpenSCAD**  
Skriptdateibasierte grafische CAD-Umgebung  
★★★★★ (4 Bewertungen)

**Installiert am 02.05.2016** Entfernen

OpenSCAD is a software for creating solid 3D CAD objects. It focuses on CAD aspects rather than artistic ones.

OpenSCAD is not an interactive modeller. Instead it is something like a 3D-compiler that reads in a script file that describes the object and renders the 3D model from this script. This gives the designer full control over the modelling process and enables him to easily change any step in the modelling process or make designs that are defined by configurable parameters.

[Entwicklerwebseite](#)

**Zusätzliche Erweiterungen**

- Geomview (geomview)
- LibreCAD (librecad) ★★★★★
- MeshLab (meshlab) ★★★★★
- Script file based graphical CAD environment (test suite) (openscad-testing)

Version openscad 2015.03-14dfsg-2ubuntu1  
Gesamtgröße 7,0 MB auf der Festplatte  
Lizenz Quelloffen

Aktualisierungen Canonical stellt keine Aktualisierungen für OpenSCAD bereit. Einige Aktualisierungen sind möglicherweise über die Ubuntu-Gemeinschaft verfügbar.

**Bewertungen** Deutsch Hilfreichste zuerst

[Eine eigene Bewertung schreiben](#)

Diese Anwendung wurde in Ihrer Sprache noch nicht bewertet.  
Wählen Sie eine andere Sprache oder »Beliebige Sprache« in der Sprachauswahl.

## FreeCAD

Ubuntu Software-Center

Alle Anwendungen | Installiert | Verlauf

## FreeCAD

Eigenschaftsbasierter parametrischer Modellierer  
★★★★☆ (38 Bewertungen)

✓ Installiert am 13.12.2015 Entfernen

FreeCAD ist ein quelloffenes CAX-RAD-System, das OpenCasCade, Qt und Python verwendet. Es unterstützt einige Schlüsselkonzepte, wie die Aufnahme von Makros, Workbenches (Werkbänke), Fähigkeit als Server zu laufen und dynamisch ladbare Anwendungserweiterungen. Es ist plattformunabhängig konzipiert.

Aktuell kann FreeCAD CAD-Modelle in den Formaten IGES, STEP und BREP und Drahtgitter in den Formaten STL, BMS, AST und Wavefront OBJ importieren und anzeigen. Die Möglichkeiten für Bearbeitung und Modellierung sind derzeit etwas begrenzt.

[Entwicklerwebseite](#)

### Zusätzliche Erweiterungen

- FreeCAD documentation (freecad-doc)

Version	freecad 0.14.3702+dfsg-4
Gesamtgröße	63,2 MB auf der Festplatte
Lizenz	Quelloffen

Aktualisierungen Canonical stellt keine Aktualisierungen für FreeCAD bereit. Einige Aktualisierungen sind möglicherweise über die Ubuntu-Gemeinschaft verfügbar.

### Bewertungen

[Eine eigene Bewertung schreiben](#)

Für die Version 0.12.5284-dfsg  
★★★★☆ **Neuere Version**  
Hier im Software-Center wird nicht die aktuellste Version installiert, diese gibt es im PPA repository.

Folgende Quelle ist hinzuzufügen:  
ppa:freecad-maintainers/freecad-stable

Stand heute 28.09.2013

Deutsch | Hilfreichste zuerst

Wolfgang, 2013-09-28

<https://www.tinkercad.com/> Make an free account and do the first 3 lessons, so you know the basics about tinkercad.

The extension, you will find here: <https://www.thingiverse.com/thing:25036>

Now copie the 2 files into your extension folder of inkscape, perhaps you need to use the command-line cp to do that. Here is an example how you use cp for it:

```
fantawams@fantawams-ThinkPad-L450:~$ sudo cp /home/fantawams/Schreibtisch/paths2openscad.inx /usr/share/inkscape/extensions
fantawams@fantawams-ThinkPad-L450:~$ sudo cp /home/fantawams/Schreibtisch/paths2openscad.py /usr/share/inkscape/extensions
```

sudo cp „from file destination“ „to folder destination“

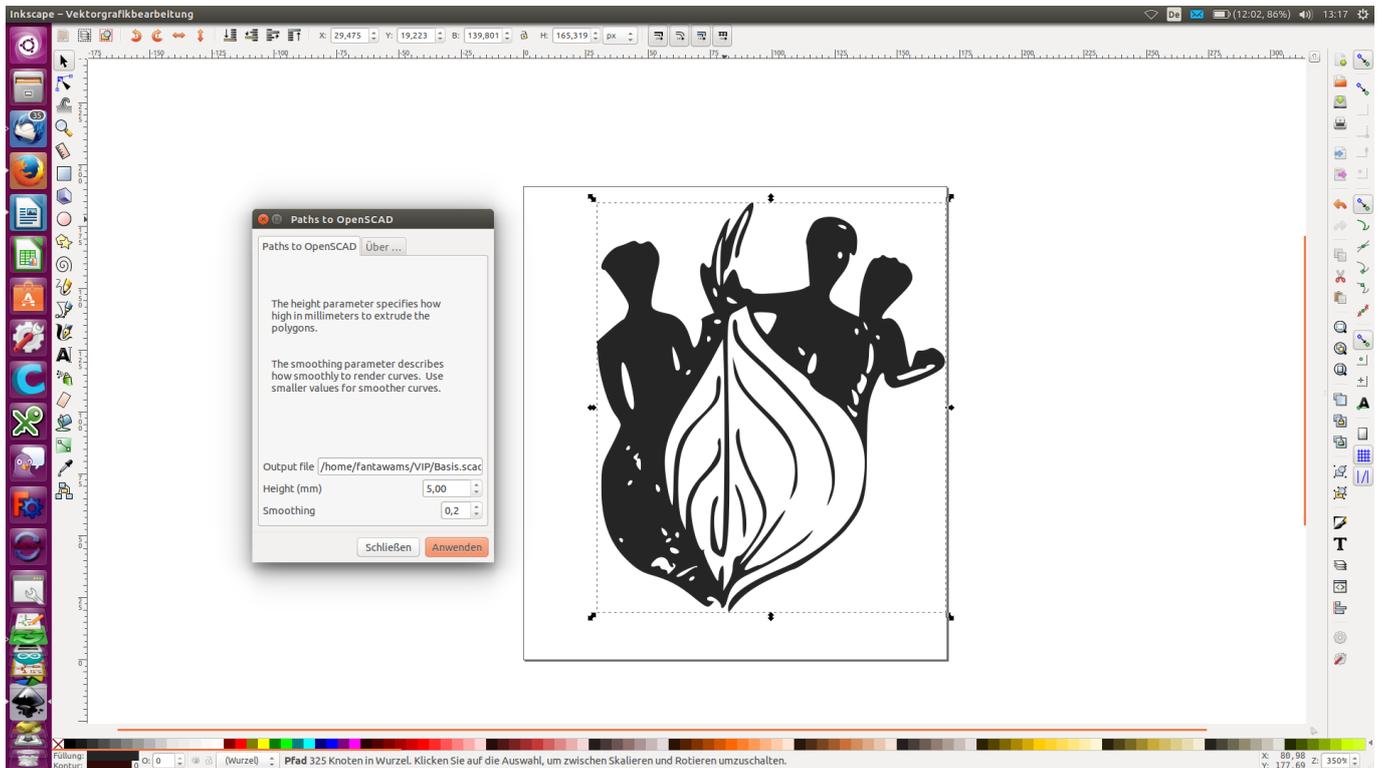
Now open Inkscape and look under extension/create out of path/ there should be something like „path to Open SCAD“. If not, than the extension is not copied correctly.

Now that your PC is prepared, let's start working with it.

## Converting your Image

First open Inkscape and load your Image, remember it is best to use an black and white image and in an SVG format.

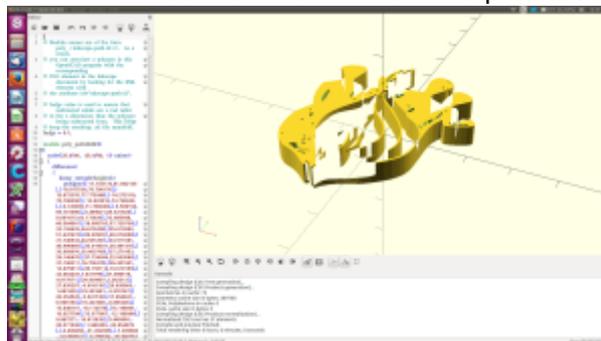
Now click one time on your Image and use your new extension.



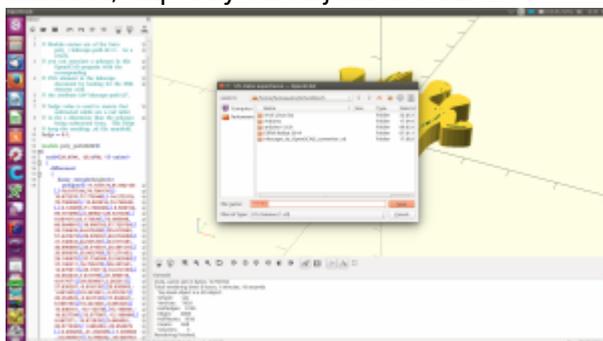
Now check if in Output file is something like this: /home/fantawams/VIP/Basis.scad. On the first time there might be an ~ before /home, if you have that delete the ~ or you won't be able to save your scad-file. Change your Hight as you want to, this determines how high or thick your peace will be . Next click OK to create an SCAD-file.

Now open your new scad-file with OpenSCAD.

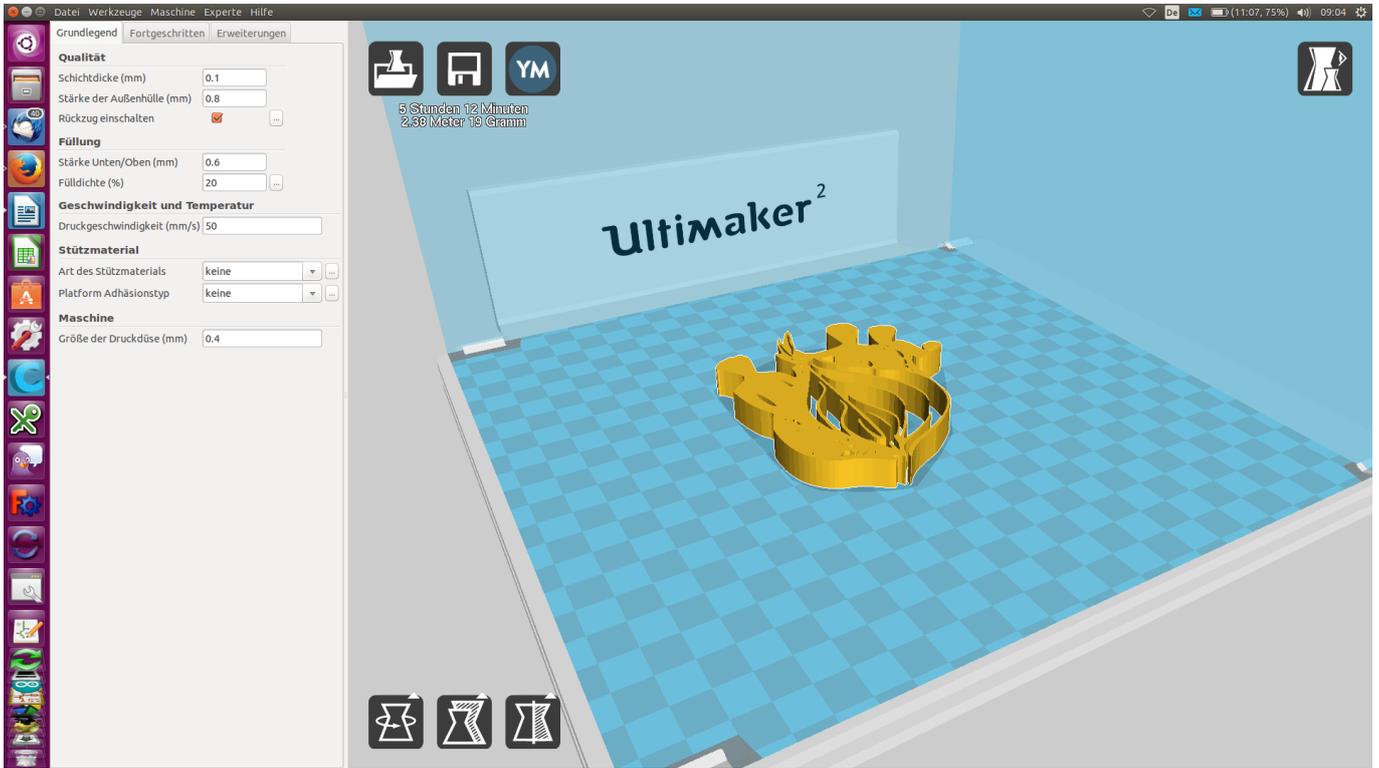
Next click on Design and Render. That will take some time to finish the process. After the process is



finished, export your object as STL.



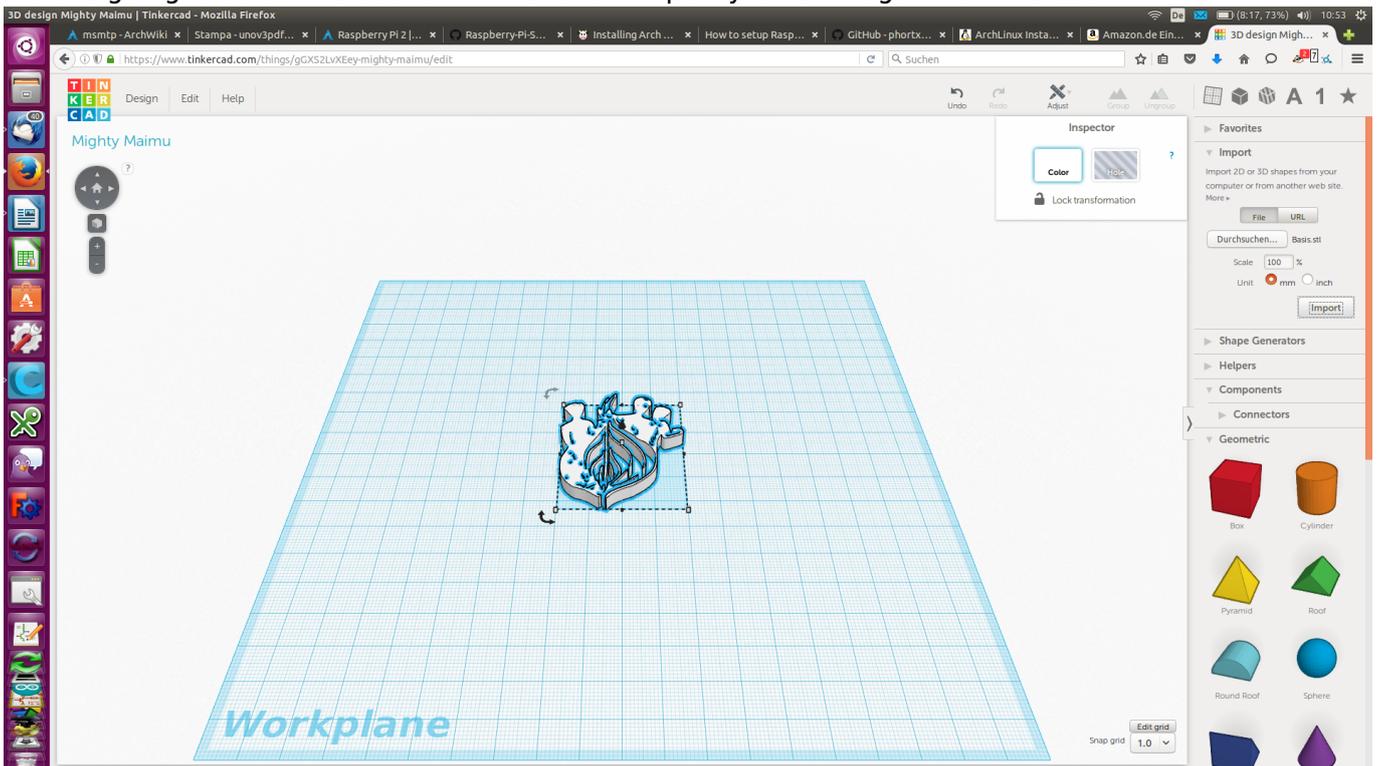
So now you have you 2D logo as an printable STL, if you like, you could print it now or you could edit it like in our next step.



## Editing your STL with tinkercad

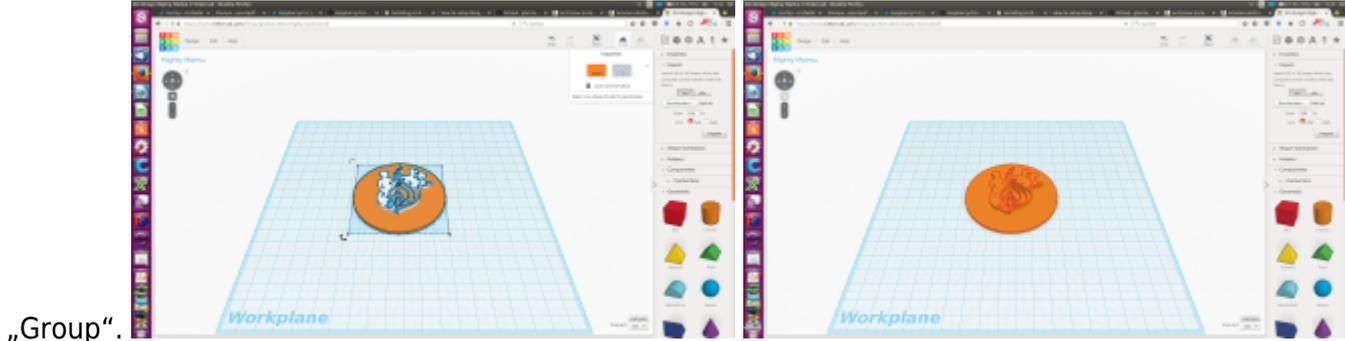
Before we start, here are 2 alternatives of editing your new STL-logo: Meshmixer: it is an very easy to use program with which you can edit your STLs like you want to, combining or making a difference is no problem for it. Blender: Blender is the ultimate tool, you can do what you want but it is very complex and complicated. So it's difficult to learn.

We are going to use tinkercad for it. First of all import your STL-logo into tinkercad.

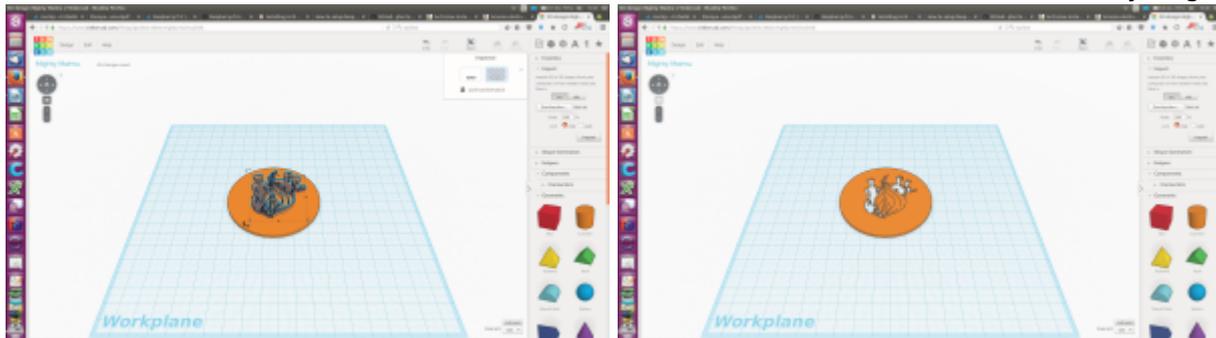


Next import an second STL which you draw in you CAD-program or draw one in tinkercad.

If you want to combine these 2 STLs, than move them together and select both of them. Than Click



If it should be an difference of both of them, than make them an „Hole Tool“ before you group them.



It is as simple as that and you're done with your new STL or if you want, you can edit some more. After all your only limited by your imagination and creativity. Now your ready to print. If you still have some questions, send me an email which you can find [here](#).

## Sources

<https://wiki.ubuntuusers.de/cp/>

From:  
<https://wiki.c3l.lu/> - **Chaos Computer Club Lëtzebuerg**

Permanent link:  
<https://wiki.c3l.lu/doku.php?id=projects:howtos:2dimage-to-3dstl>

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